

FLOW AND HEAT TRANSFER IN 180-DEGREE TURN SQUARE DUCTS
- EFFECTS OF TURNING CONFIGURATION AND SYSTEM ROTATION

Ten-See Wang
Computational Fluid Dynamics Branch
NASA - Marshall Space Flight Center
Marshall Space Flight Center, AL 35812

Ming-King Chyu
Department of Mechanical Engineering
Carnegie Mellon University
Pittsburgh, PA 15213

521-34
p. 19

ABSTRACT

Forced flow through channels connected by sharp bends is frequently encountered in various rocket and gas turbine engines. For example, the transfer ducts, the coolant channels surround the combustion chamber, the internal cooling passage in a blade or vane, the flow path in the fuel element of a nuclear rocket engine, the flow around a pressure relieve valve piston, and the recirculated base flow of multiple engine clustered nozzles. Transport phenomena involved in such a flow passage are complex and considered to be very different from those of conventional turning flow with relatively mild radii of curvature. While previous research pertaining to this subject has been focused primarily on the experimental heat transfer, very little analytical work is directed to understanding the flowfield and energy transport in the passage. Therefore, the primary goal of this paper is to benchmark the predicted wall heat fluxes using a state-of-the-art computational fluid dynamics (CFD) formulation against those of measurement for a rectangular turn duct. Other secondary goals include studying the effects of turning configurations, e.g., the semi-circular turn, and the rounded-corner turn, and the effect of system rotation. The computed heat fluxes for the rectangular turn duct compared favorably with those of the experimental data. The results show that the flow pattern, pressure drop, and heat transfer characteristics are different among the three turning configurations, and are substantially different with system rotation. Also demonstrated in this work is that the present computational approach is quite effective and efficient and will be suitable for flow and thermal modeling in rocket and turbine engine applications.

PRECEDING PAGE BLANK NOT FILMED

Flow and Heat Transfer in 180-Degree Turn Square Ducts - Effects of Turning Configuration and System Rotation

Ten-See Wang

**Computational Fluid Dynamics Branch
NASA-Marshall Space Flight Center**

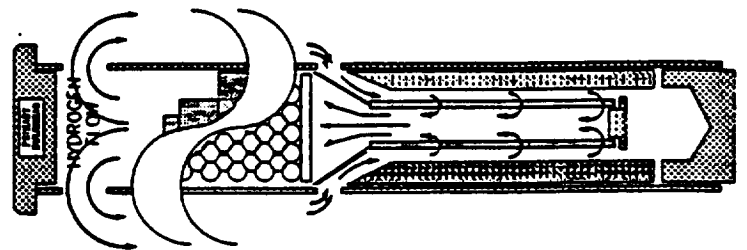
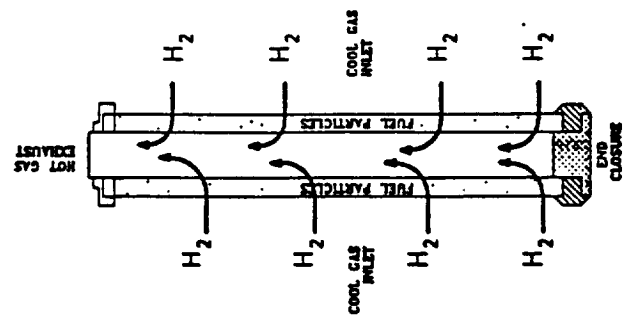
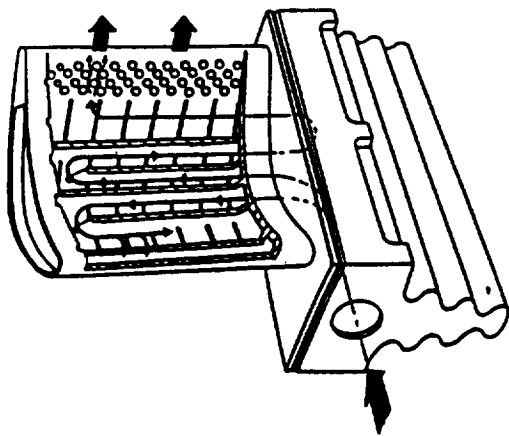
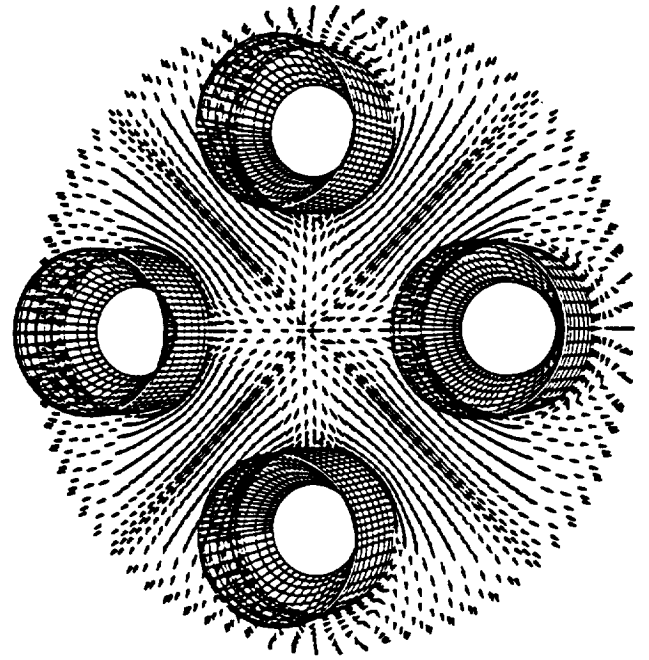
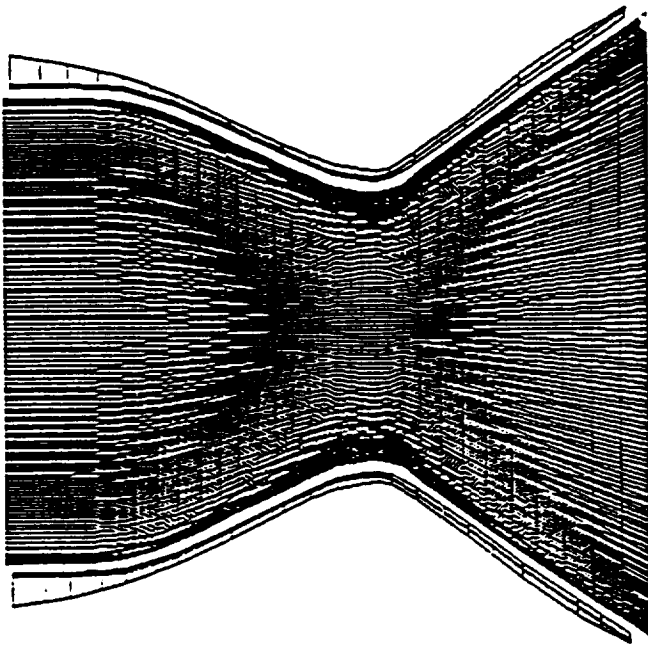
Ming-King Chyu

**Department of Mechanical Engineering
Carnegie Mellon University**

**11th Workshop for CFD in Rocket Propulsion
Main Sessions 5: Applications - Duct Flows**

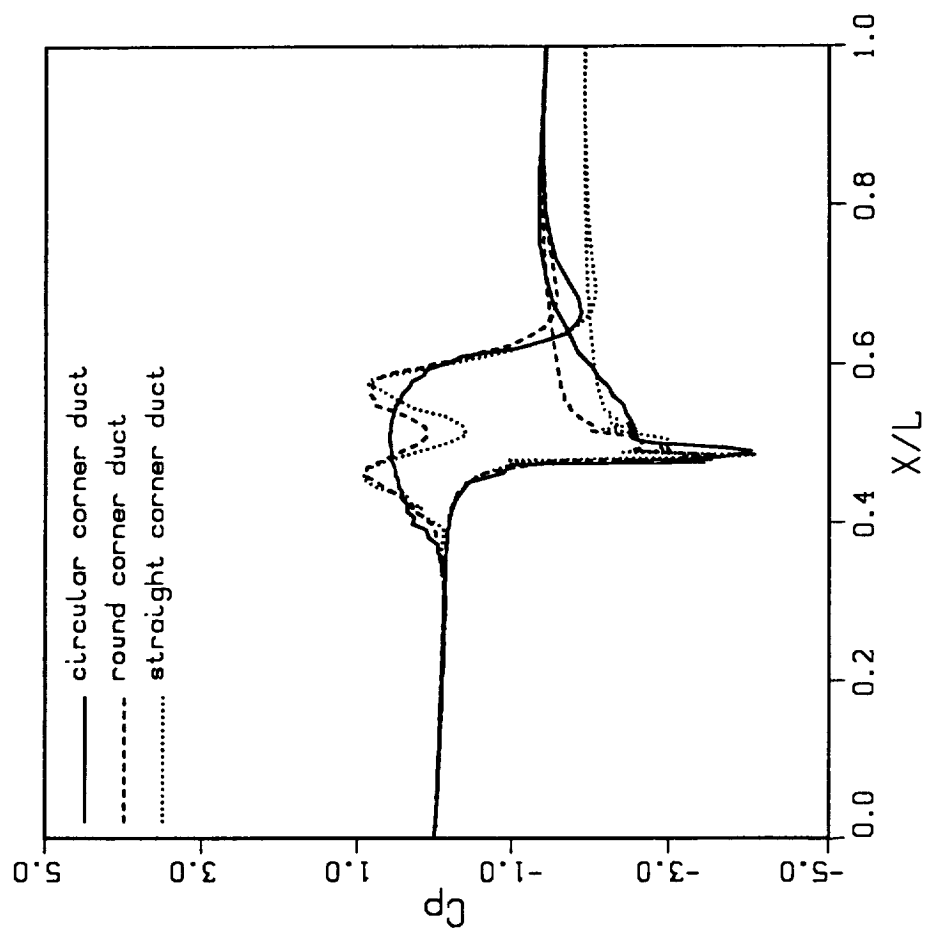
MSFC, Alabama

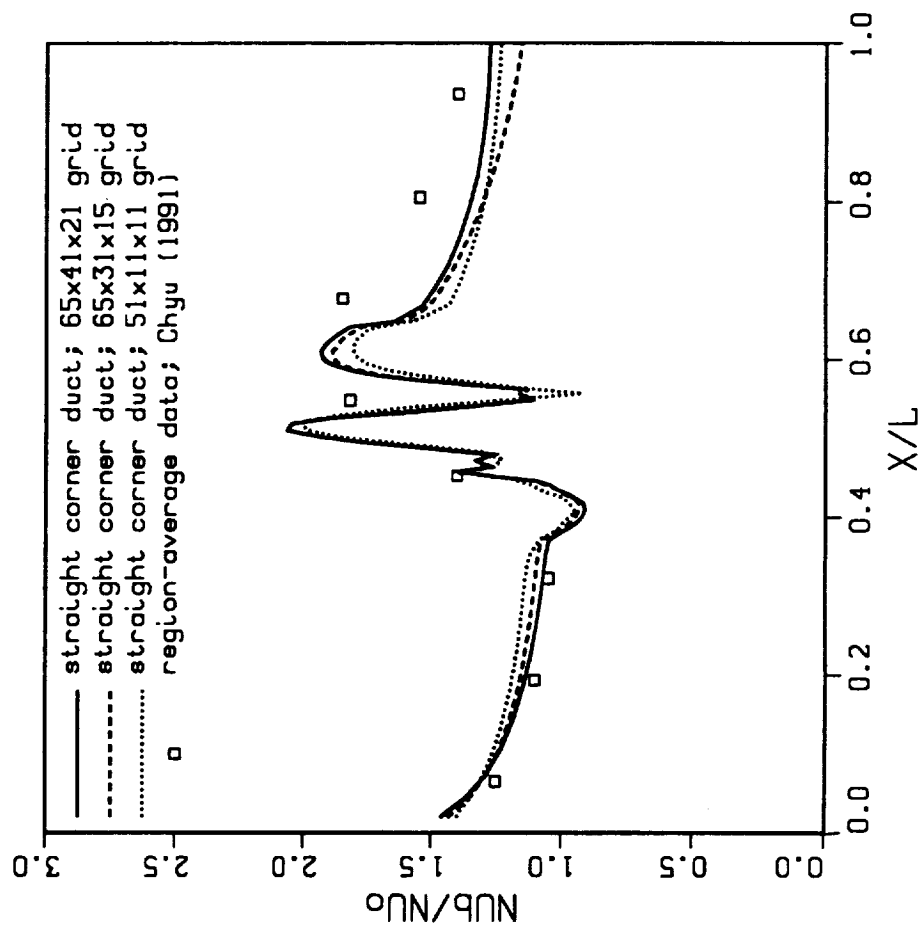
April 20, 1993

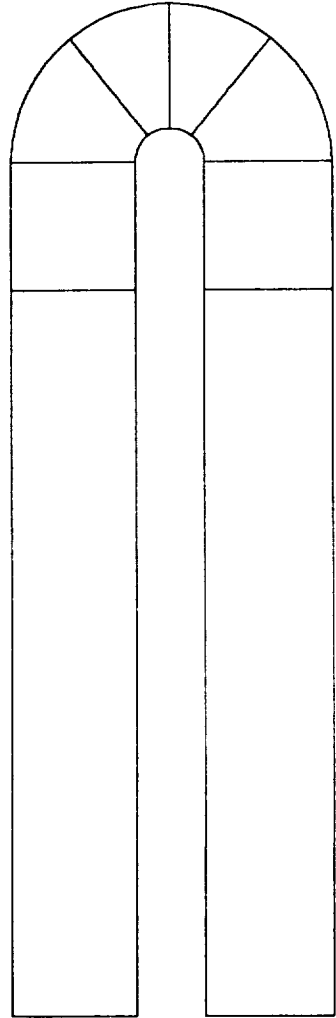
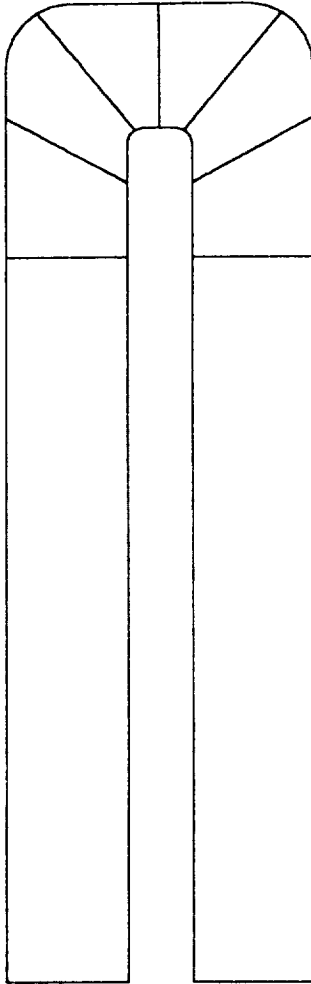
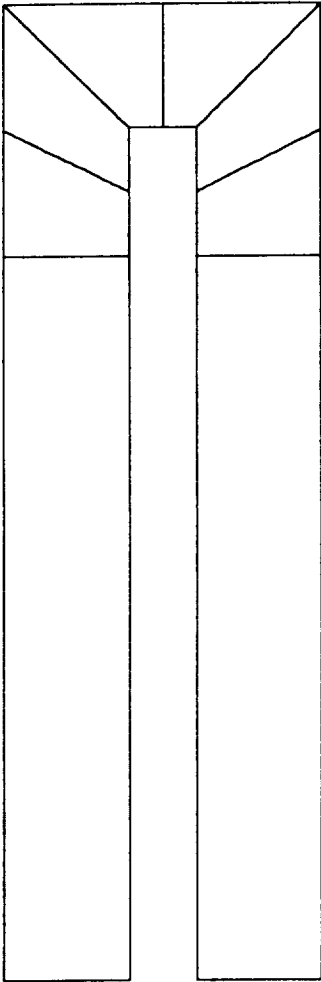


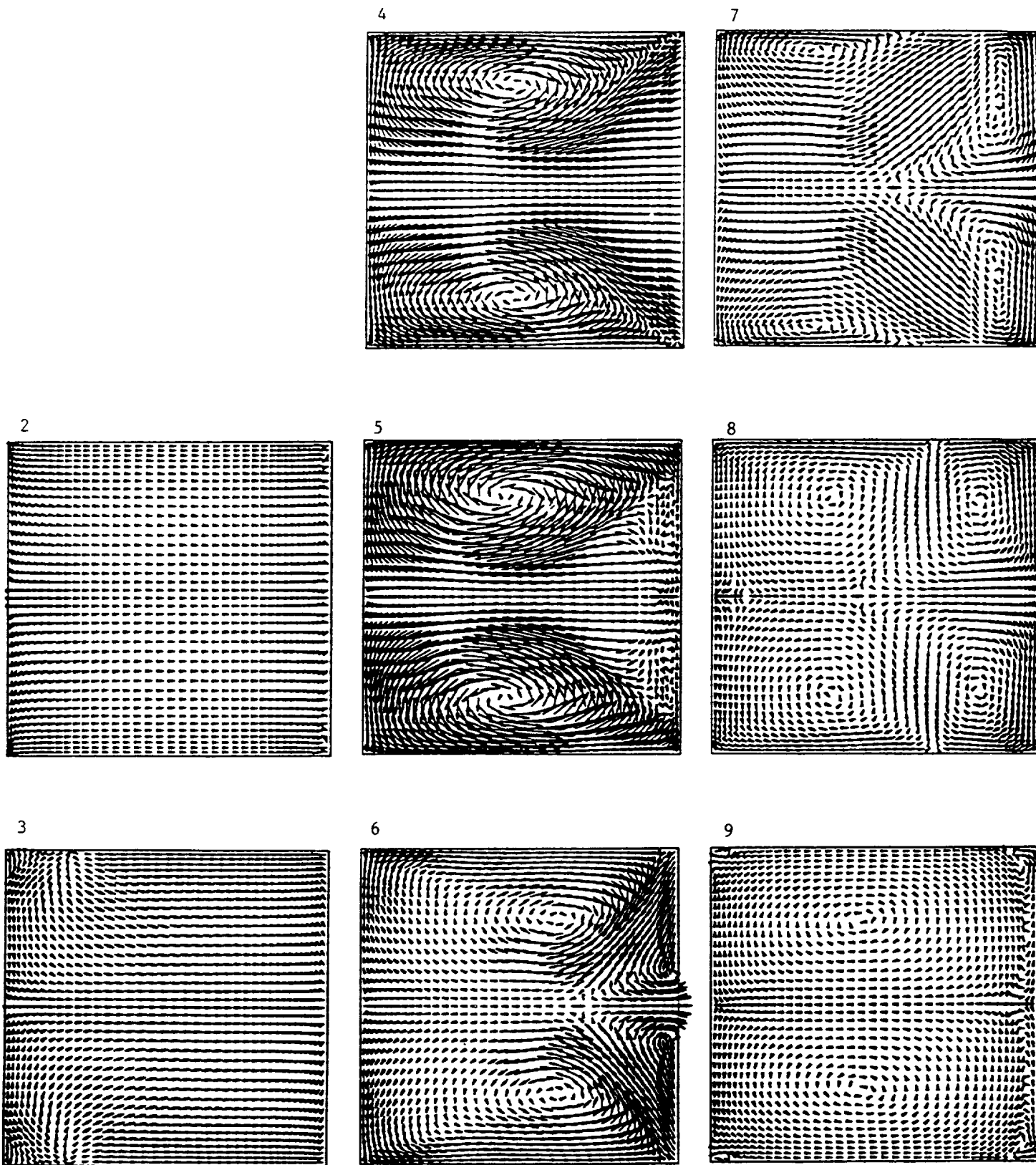
Objectives

- To obtain detailed information on the turbulent flowfield and its effects on the local heat transfer
- To benchmark the CFD solution with a two-pass sharp-turned square duct experiment
- To examine the influence of turn configuration and System rotation on the overall transport phenomena in the passage

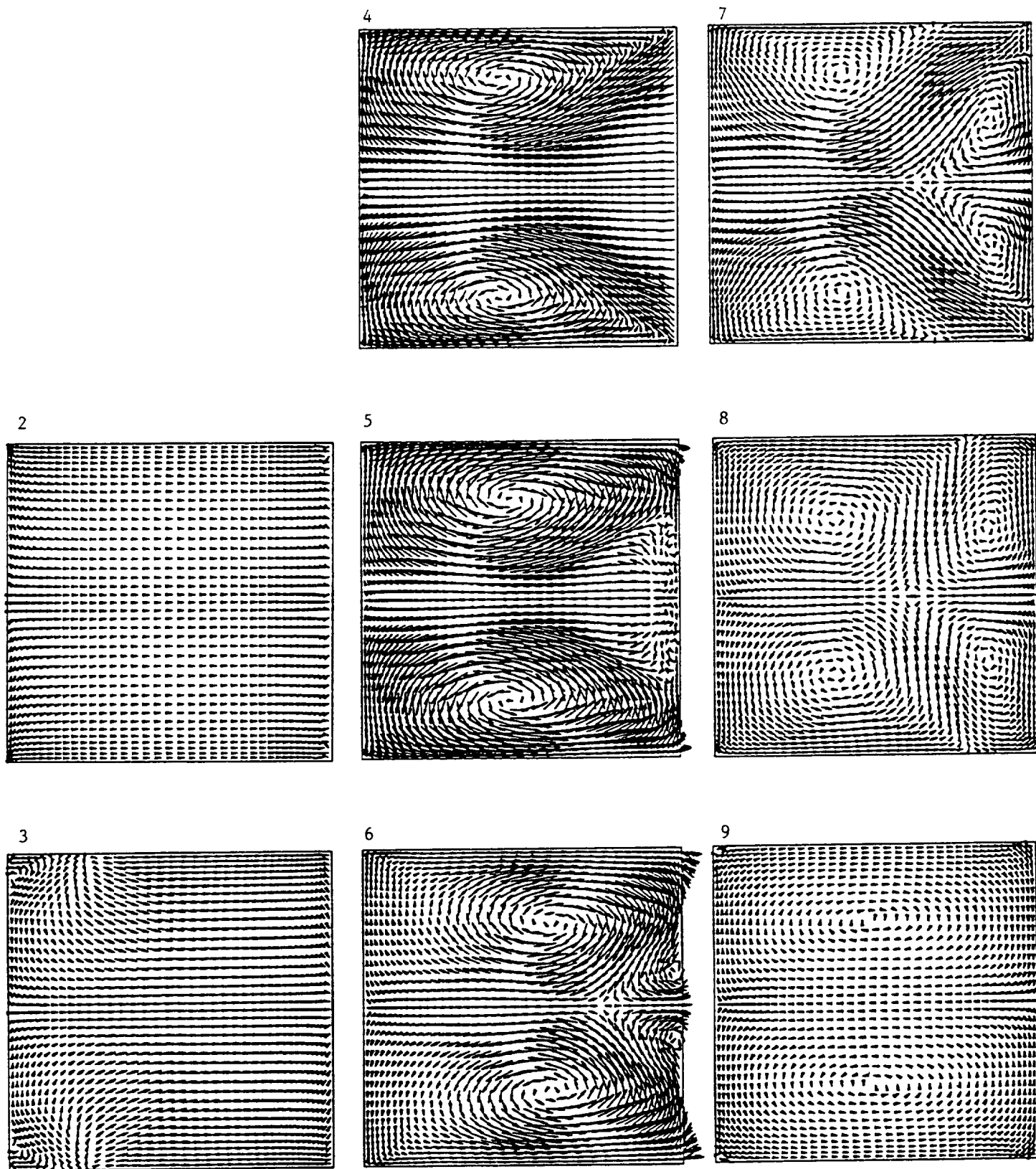




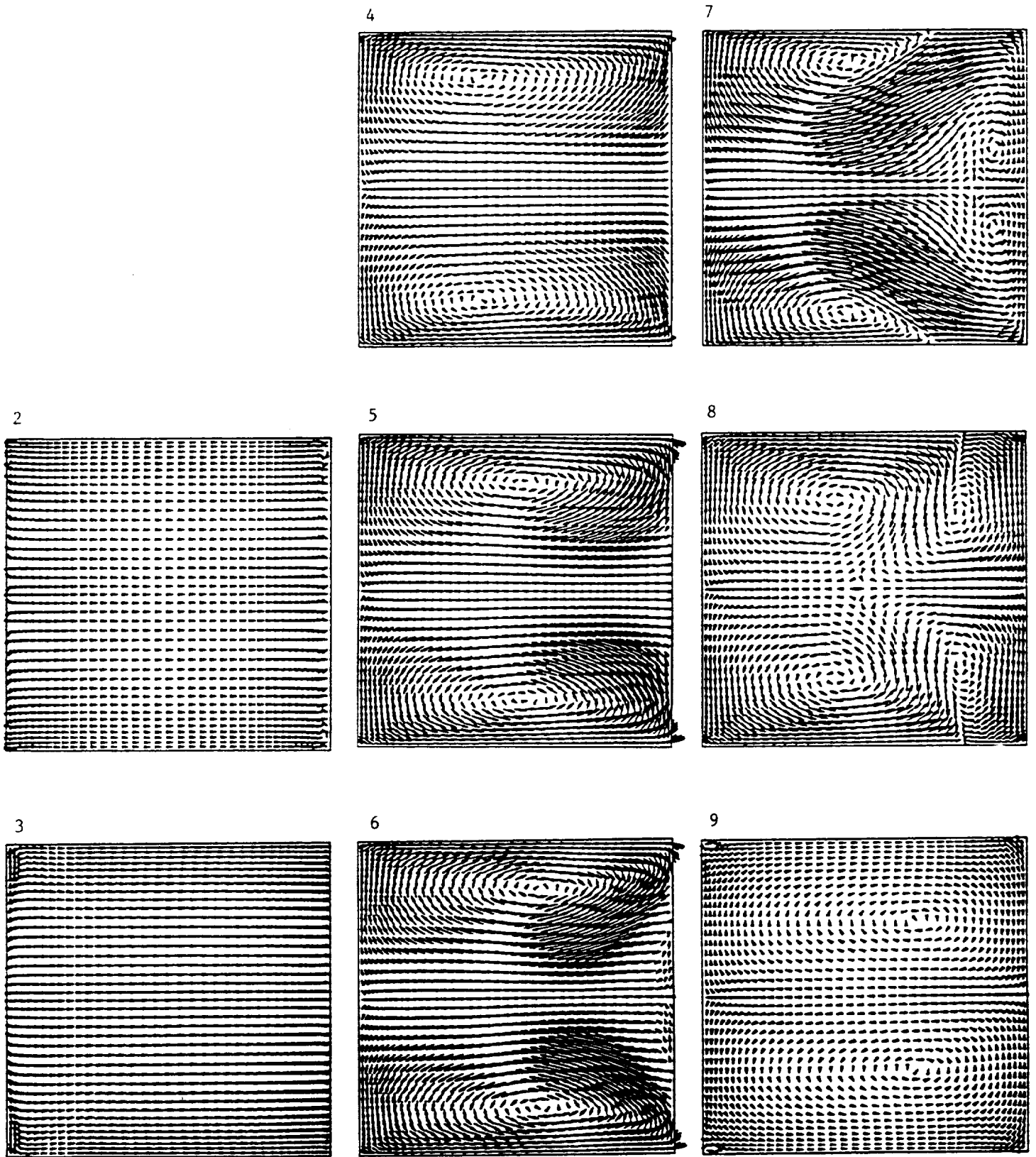




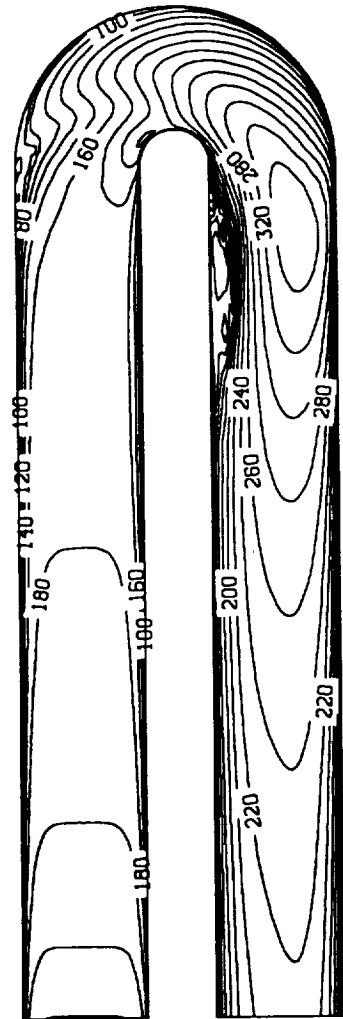
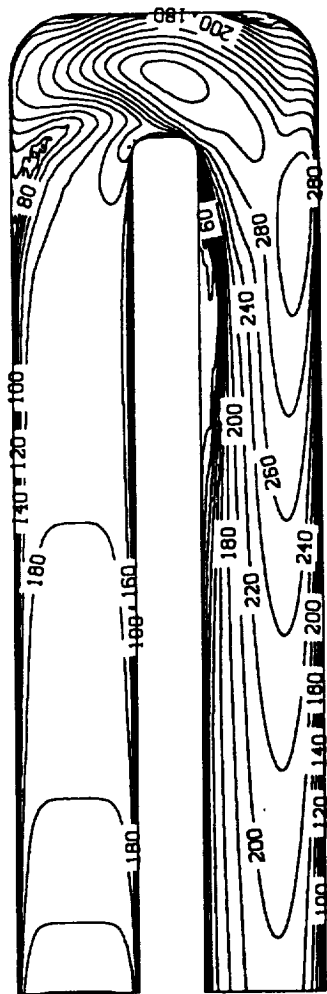
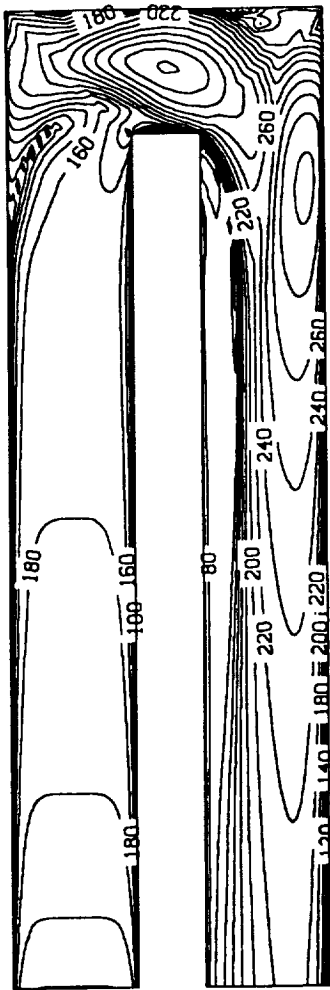
The evolution of the secondary flow pattern for a 180-degree turn straight-corner square duct

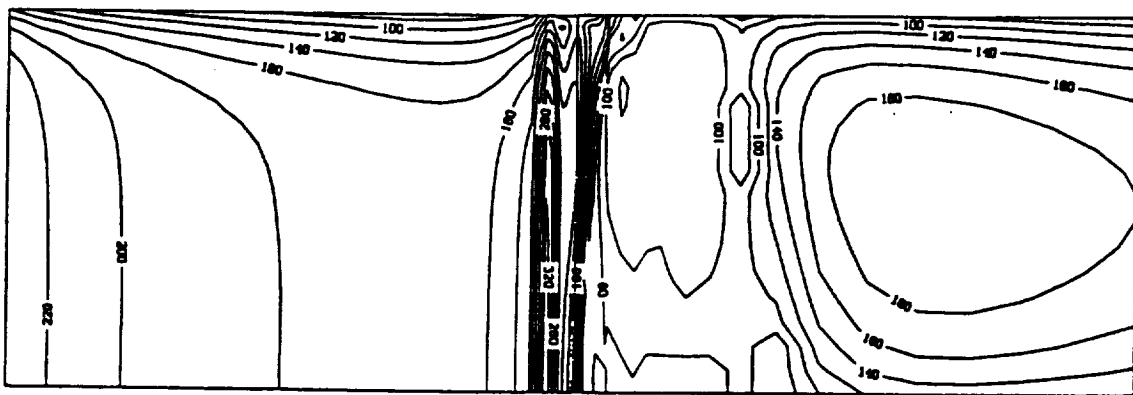
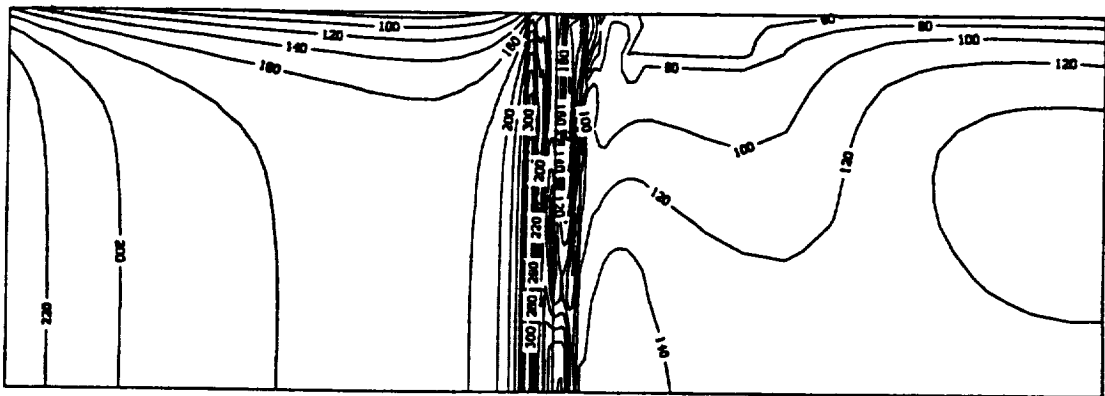
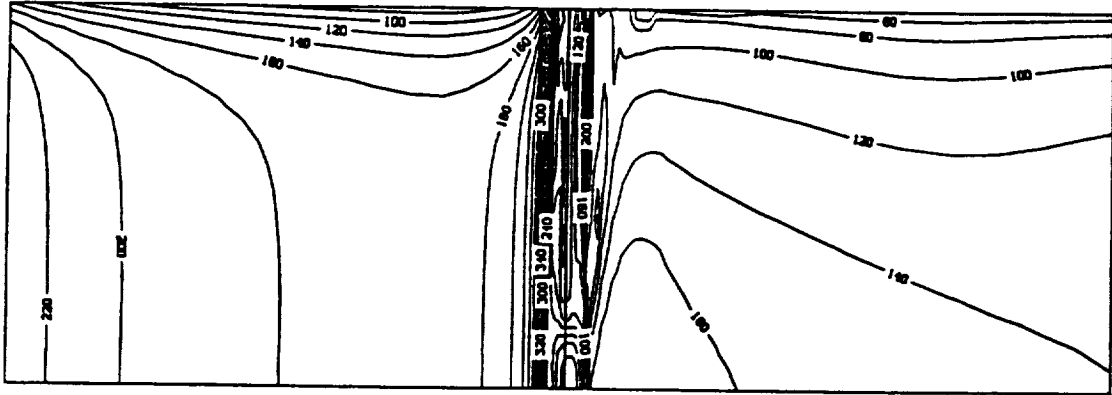


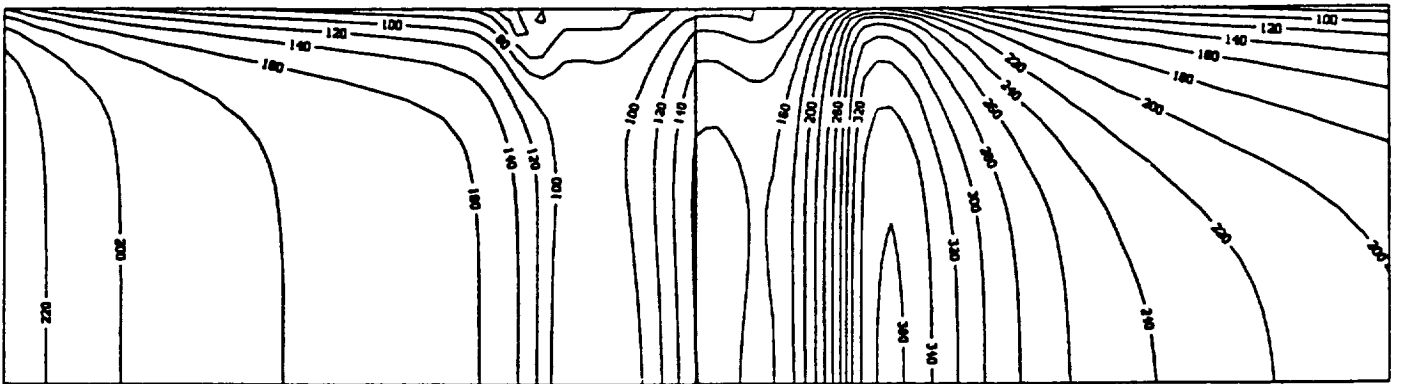
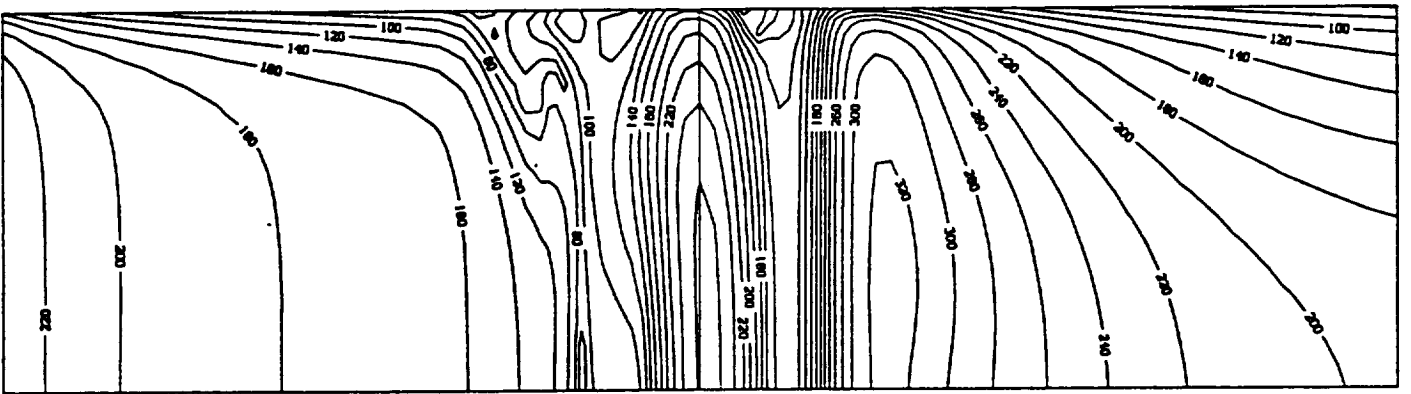
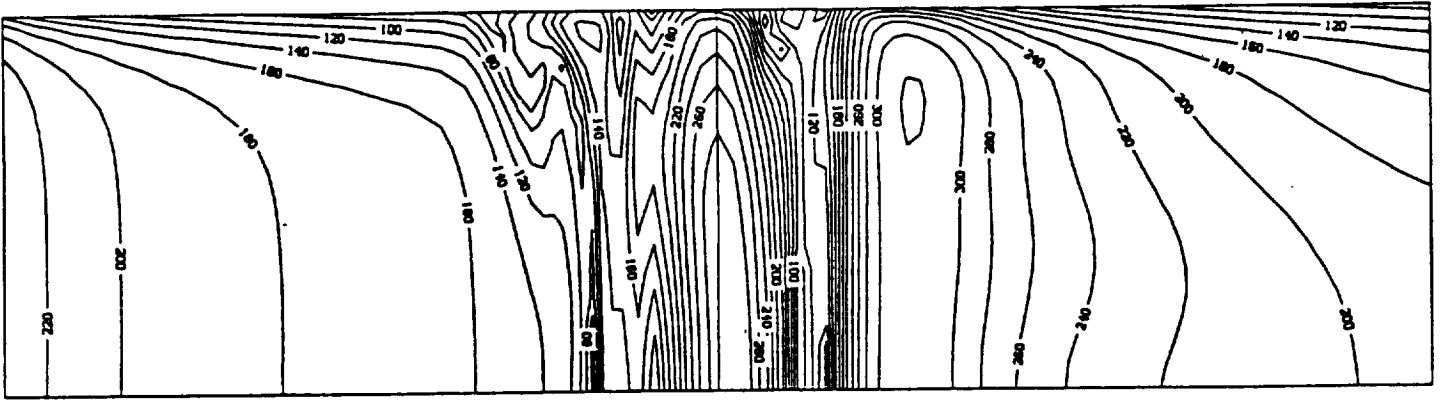
The evolution of the secondary flow pattern for a 180-degree turn round-corner square duct

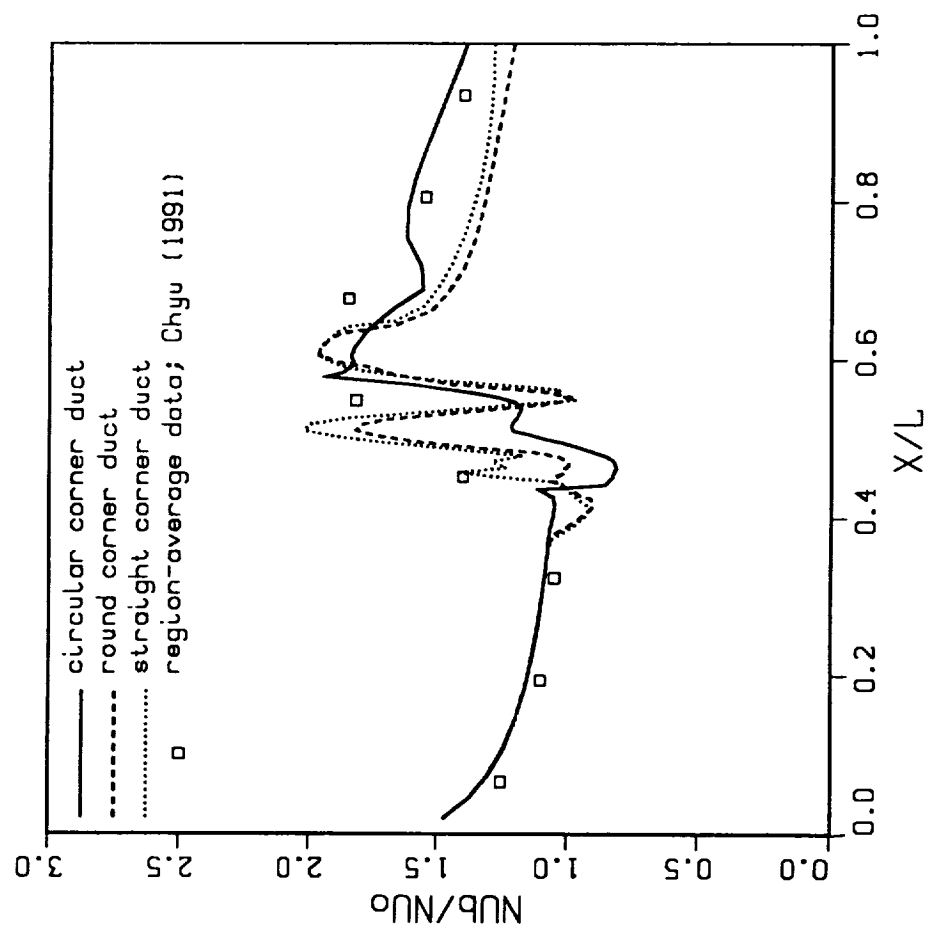


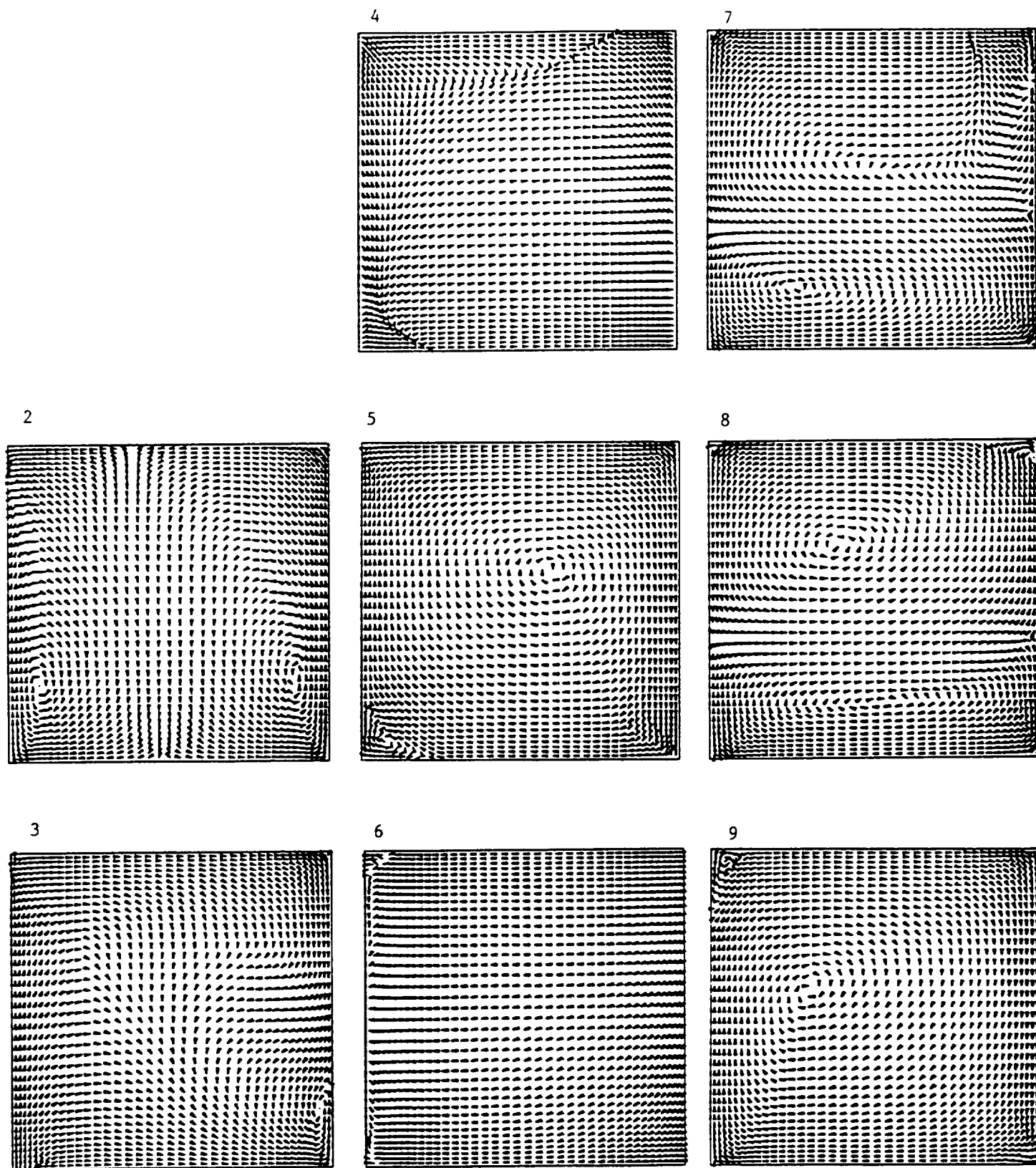
The evolution of the secondary flow pattern for a 180-degree turn circular-corner square duct



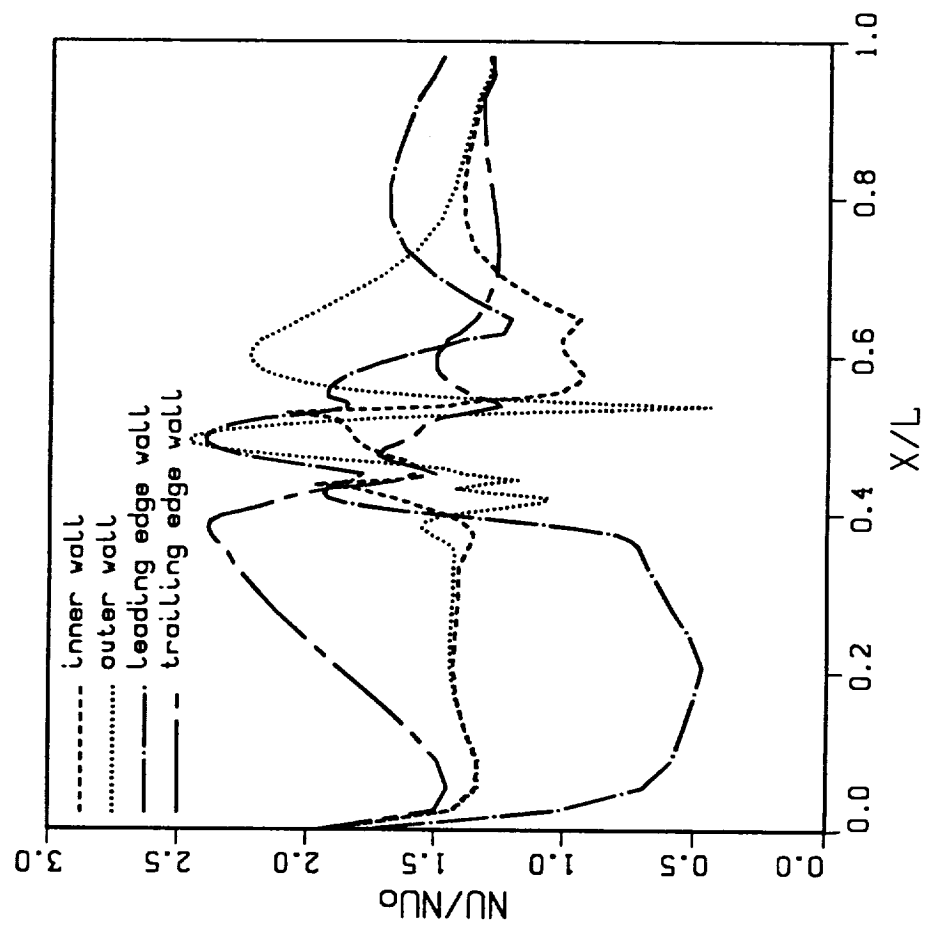


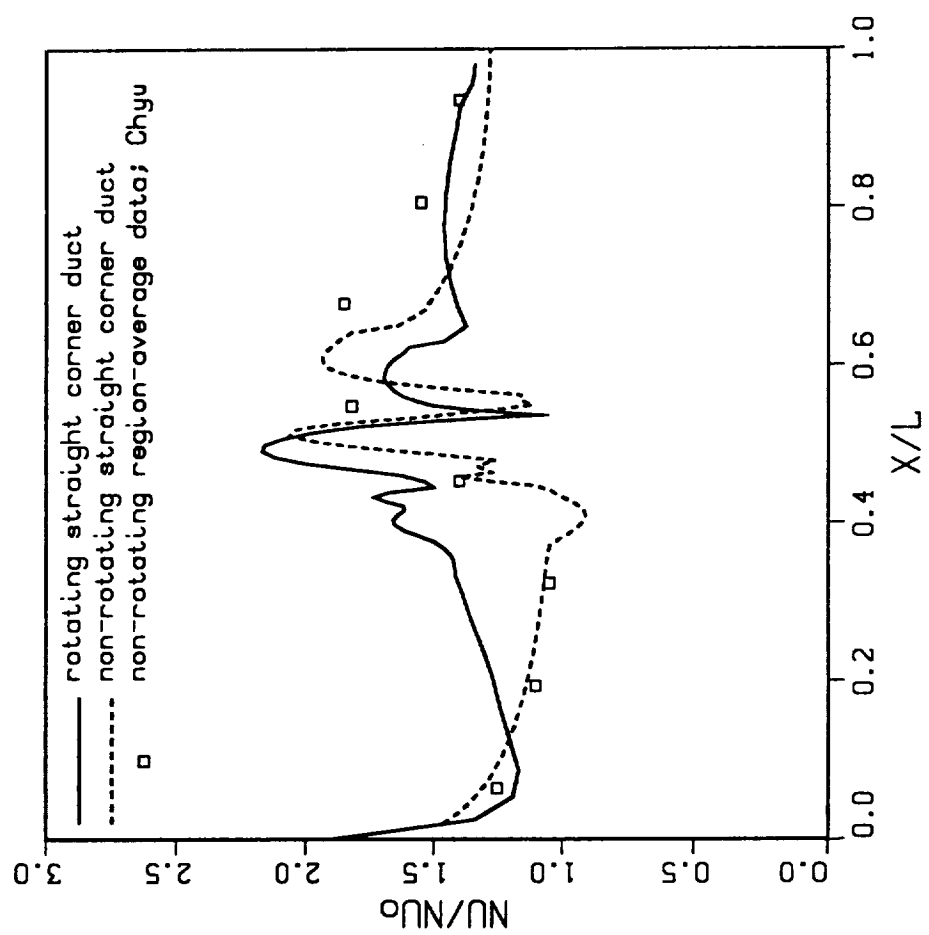






The evolution of the secondary flow pattern for a 180-degree turn straight-corner square duct with rotation





The averaged total Nusselt numbers

	NUi	NUb	NUb/NUo
Circular-Corner Turn	184	210	1.3127
Round-Corner Turn	188	213	1.3321
Straight-Corner Turn	193	222	1.3837
Straight-Corner Turn with Rotation	217	250	1.5614

Conclusions

- Detailed information in flowfield, pressure drop and local heat transfer
- Favorable agreement with experimental data
- Complex features and strong turn-geometry dependence in post-turn secondary flows
- Straight-corner turn has highest turn-induced heat transfer enhancement
- Circular-corner turn has highest post-turn heat transfer
- System rotation enhances heat transfer

